DEC STD 114-1 Engineering Drawing Requirements - Dimensioning and Tolerancing for Engineering Drawings

DOCUMENT IDENTIFIER: A-DS-EL00114-01-0000 Rev B1, 15-Jun-1990

ABSTRACT: This standard presents Digital's requirements for dimensioning and tolerancing engineering drawings. It provides guidelines for using inches and millimeters as units of measure, and for application of general tolerancing on engineering drawings.

APPLICABILITY: Mandatory for all engineering and manufacturing organizations that prepare and use engineering drawings.

STATUS: APPROVED 15-Jun-1990; use VTX SMC for current status.

This document is confidential and proprietary, and is the property of Digital Equipment Corporation. It is an unpublished work protected under the Federal copyright laws.

©Digital Equipment Corporation. 1990. All rights reserved.

DEC STD 114-1 Engineering Drawing Requirements - Dimensioning and Tolerancing for Engineering Drawings

DOCUMENT IDENTIFIER: A-DS-EL00114-01-0000 Rev B1, 15-Jun-1990

REVISION HISTORY:	Rev A,	07-Oct-1982	
	Rev B,	23-Feb-1989	ECO Number CT001
	Rev B1,	15-Jun-1990	ECO Number CTS02

Document Management Category:	Engineering Design/Documentation Methods (HPD)
Responsible Department:	Engineering Support Operations
Responsible Person:	Phil Reed
SMC Writer:	Gloria Chao

APPROVAL: This standard has been reviewed and recommended for approval by the General Review group for its category. Rev B1 incorporates clerial changes only.

Phil Reed, Standard Owner

Eric Williams - Standards Process Manager

Direct requests for further information to:

Phil Reed
Use $ VTX ELF for the latest location information.

Use VTX SMC to order copies of this document from Standards and Methods Control. Send distribution questions to JOKUR::SMC or call DTN: 234-4423.

CONTENTS

1 INTRODUCTION 1
1.1 PURPOSE 1
1.2 SCOPE 1
1.3 RESPONSIBILITIES 1
1.3.1 Responsible Product Engineer 1

2 GENERAL REQUIREMENTS 1

3 UNITS OF MEASURE FOR DIMENSIONING AND TOLERANCING 2
3.1 U.S. CUSTOMARY (INCH) DIMENSIONING 2
3.2 METRIC DIMENSIONING 2
3.3 DUAL DIMENSIONING 3

4 DECIMAL DIMENSIONING 3

5 TOLERANCING 4
5.1 NUMBER OF DECIMAL PLACES SENSITIVE 4
5.2 DIMENSION SENSITIVE 4

6 FRACTIONAL DIMENSIONING 5
6.1 NEW PARTS DOCUMENTATION 5
6.2 EXISTING PARTS DOCUMENTATION 5

7 TITLE BLOCKS 6

8 CONVERTING INCH DIMENSIONS TO METRIC 9

9 CONVERTING FRACTIONS TO DECIMAL 9

Appendix A REFERENCED DOCUMENTS 11

A.1 EL-Class Digital Documents 11

A.2 Other Documents 11

A.3 Ordering Information 11

FIGURES

1 Tolerances on Drawing 4
2 Tolerances in Text 5
3 Limits on Drawing 5
4 Limits in Text 5
5 Standard Digital U.S. Customary (Inch) Title Block 7
6 Standard Digital Metric Title Block 7
7 First Option Selected 8
8 Third Option Selected 8

TABLES

1 Requirements Comparison
ANSI Y14.5-1973 References Versus Digital 2

1 INTRODUCTION

1.1 PURPOSE

This standard presents Digital's requirements for representing dimensions and dimension tolerances on engineering drawings.

1.2 SCOPE

Dimensions on engineering drawings may be represented by either U.S. customary (inch) dimensions, metric, or dual (inch and metric) dimensions.

This standard describes the correct method for specifying either type of dimension.

This standard also describes the U.S. customary (inch) and metric tolerance blocks used on pre-printed engineering drawing forms, and explains how to use them.

This standard does not specify when to use U.S. customary dimensions or metric dimensions; that decision is made by the mechanical engineer who is responsible for the engineering drawings for each product. The only general rule that can be applied is to use dual dimensioning on any engineering drawings that may be provided to customers.

This standard supersedes *DEC STD 019-0 Decimal Dimensioning Standard* and *DEC STD 114 Metric Dimensioning on Engineering Drawings, dated September 1974*.

1.3 RESPONSIBILITIES

1.3.1 Responsible Product Engineer

The Product Engineer is responsible for determining if the design, manufacture, or maintenance of the product requires inch, metric, or dual dimensioning on its documentation. This individual is also responsible for ensuring that these requirements are implemented by internal and external support organizations.

2 GENERAL REQUIREMENTS

Primarily, Digital has adopted ANSI Y14.5M-1982 as its standard for dimensioning and tolerancing practice, supplemented by Geometric Dimensioning and Tolerancing - A Working Guide by Lowell Foster for Geometric dimensioning applications.

In certain areas of engineering documentation, Digital has other standards that supplement or replace those specified by ANSI Y14.5M-1982. The standards specified by ANSI Y14.5M-1982 and those available within Digital are listed for various documentation areas in Table 1-1.

Third angle projection shall be used throughout Digital; European first angle projection shall not be used.

3 UNITS OF MEASURE FOR DIMENSIONING AND TOLERANCING

3.1 U.S. CUSTOMARY (INCH) DIMENSIONING

For linear measurements on U.S. products, Digital uses U.S. Customary inches and decimal parts of an inch.

For inch measurements less than 1, omit zeroes left of the decimal point.

Dimensions and tolerances to four decimal places should be avoided if physical inspection is involved, because normal quality control practice requires gauges 10 times as accurate as the dimension to be measured, and gauges with 5-place accuracy are expensive and difficult to obtain.

Angular dimensions and tolerances shall be specified in degrees and, where necessary, in minutes and seconds. An angular dimension and both its tolerance limits shall use the same units of measure.

Table 1: Requirements Comparison ANSI Y14.5-1973 References Versus Digital

Subject	ANSI Y14.5	Digital
Drawing sheet size and format	Y14.1-1975	DEC STD 013-1
Decimal inch dimensioning	B87.1-1965	DEC STD 114
Rules for rounding numbers	B87.1-1965	DEC STD 114
Line conventions	Y14.2M-1979	Y14.2M-1979
Lettering	Y14.2M-1979	DEC STD 182
Knurling	B94.6-1981	B94.6-1981
Screw threads	Y14.6-1957	Y14.6-1957
Surface texture	B46.1-1978	B46.1-1978
Gears, splines, and serrations	Y14.7-1958	Y14.7-1958
Spur helical, double helical, and rack gear	Y14.7.1-1971	Y14.7.1-1971
Metric dimensioning	Y14.5-1973	This section and ANSI Y14.5-1973
Dual dimensioning	Y14.5-1973	This section and ANSI Y14.5-1973

3.2 METRIC DIMENSIONING

For metric linear dimensions, Digital uses millimeters and decimal parts of a millimeter.

The symbol Ø means diameter, and is to be used on all metric drawings. The term "dia" is not an acceptable abbreviation of diameter.

Until an acceptable metric fastener is selected, standard inch fasteners currently used for inch designs shall also be used on metric designs. Any deviations shall be reviewed by the mechanical engineer responsible for the particular product.

Angular dimensions and tolerances shall be specified in degrees and, where necessary, in minutes and seconds. The angular dimension and both its limits shall use the same units of measure.

3.3 DUAL DIMENSIONING

Dual dimensioning provides both inch and metric measurements on the same drawing. This method is used for products that are intended for both U.S. and non-U.S. markets. The preferred method of dual dimensioning is to draw a short horizontal line, and to place the metric measurement above the line and the inch measurement below the line.

For inch measurements less than 1, omit all zeroes left of the decimal point.

For other metric quantities (mass, force, pressure, stress, and so forth), refer to *DEC STD 015-0 Abbreviations and Units of Measurement*.

The symbol Ø means diameter, and is to be used on all metric drawings. The term "dia" is not an acceptable abbreviation of diameter.

Standard inch fasteners presently being used on inch designs shall also be used on dual-dimension designs. All deviations shall be reviewed by the mechanical engineer responsible for the particular product.

Angular dimensions and tolerances shall be specified in degrees and, where necessary, in minutes and seconds. An angular dimension and both its limits shall use the same units of measure.

4 DECIMAL DIMENSIONING

Decimal dimensioning shall be used for all new product drawings, and is generally preferred for all drawings. Decimal representation of the inch and millimeter both obey the following rules:

a. For decimal place sensitive dimensions and tolerances, use:
1-place decimal when the tolerance need not exceed ±0.1,
2-place decimal when the tolerance need not exceed ±0.02,
3-place decimal when the tolerance need not exceed ±0.005.

NOTE

If a tighter tolerance is required, use 4-place decimal and specify that tolerance in the body of the drawing.

b. When a dimension and its tolerance are decimal place sensitive, the dimension and both limits shall all have the same number of digits to the right of the decimal point. For a dimension-sensitive tolerance, having an equal number of digits is not significant.

c. For unilateral tolerances (one limit zero), both limits shall be specified.

d. For a diameter, the least significant digit should be an even number if feasible, so that halving it does not change the implied tolerance level.

e. Tolerances used with dimension lines shall show the tolerances following the dimension, with the plus tolerance 1/2 line up and the minus tolerance 1/2 line down, so that the minus tolerance is directly under the plus tolerance, as shown in Figure 1.

f. Tolerances specified in text shall be shown all on one line, with the plus tolerance following the dimension and preceding the minus tolerance, as shown in Figure 2.

g. Limit dimensions used with dimension lines for external features shall show the maximum value listed above the minimum value; limit dimensions for internal features shall show the minimum value listed above the maximum value. To avoid legibility problems on microfilm, the upper and lower limits shall not be separated with a dimension line or separator bar. Figure 3 illustrates this configuration.

h. Limit dimensions specified in text shall be shown on a single line, separated by a dash. The larger shall be first for external features, and the smaller shall be first for internal features. Figure 4 illustrates this configuration.

i. Decimal points shall be uniform, dense, and large enough to be clearly visible, in accordance with the requirements of *DEC STD 182-0 Engineering Documentation Acceptance Criteria*.

j. Numbers shall not contain embedded commas (often used to set off thousands).

k. Nominal designations, such as thread size, wire gauge, and sheet metal thickness, shall remain as stated, and shall not be subjected to units conversion.

l. Surface finishes shall be specified in microinches on both inch and metric drawings. Preferred values are 1000, 500, 250, 125, 63, 32, and 16 microinches.

5 TOLERANCING

Digital uses three different tolerance methods, one of which is number of decimal places sensitive; the other is dimension sensitive. Another is geometric tolerencing specified in ANSI Y14.5-1982. Which is used for a particular drawing depends on the application.

5.1 NUMBER OF DECIMAL PLACES SENSITIVE

This type of tolerance is concerned with the number of digits to the right of the decimal point. It is straightforward in application, but it is difficult to keep the tolerances meaningful when converting between inches and millimeters, or when making a major change in a dimension.

5.2 DIMENSION SENSITIVE

Dimension sensitive tolerance is well adapted to converting from inch dimensioning to metric dimensioning or the internationally required dual dimensioning.

Dimensioning sensitive tolerance is based on the SI (International Standards Organization) system of tolerances. The tolerances depend on the value of the associated dimension and the criterion of maintaining tolerance as the dimension increases. This method is preferred where constant design accuracy is desired.

Figure 1: Tolerances on Drawing

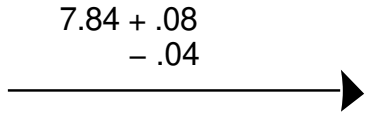


Figure 2: Tolerances in Text

7.84+.08–.04

Figure 3: Limits on Drawing

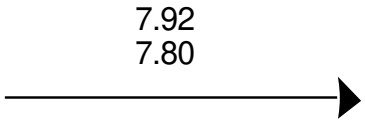


Figure 4: Limits in Text

7.92 – 7.80

6 FRACTIONAL DIMENSIONING

6.1 NEW PARTS DOCUMENTATION

Fractions are not allowed on new parts documentation because of the difficulty in making accurate conversions between decimal, fractional, and metric units, and because fractional notation can cause legibility problems when the documentation is reproduced from microfilm.

Fractions may be used on drawings that do not pertain to hardware parts or products, such as plant engineering floor plans or other drawings that:

a. Will not be distributed or reproduced on microfilm.

b. Will not require dual dimensioning.

Commercial tools and standard hardware sizes that represent measurable dimension commonly expressed as fractions shall either be specified in the decimal equivalent of the tool and gauge, or as maximum/minimum limits that include the fractional dimension and allowable tolerances. Such fractions shall be converted to decimals by use of the method described under heading 9.

Nominal sizes expressed as fractions, such as 1/2-inch iron pipe (which nowhere measures 0.5 inch), shall remain as fractions.

6.2 EXISTING PARTS DOCUMENTATION

If fractional dimensions are shown on current drawings of existing parts, they shall be converted to decimal notation when the drawings are revised by an ECO. Although exact conversion of the fractions to decimal may require 3 or 4 places, such precision may not be actually needed; the Design Engineer should reassess the actual requirements and specify the decimal precision accordingly.

7 TITLE BLOCKS

Dimensional tolerances for a drawing are determined by the design requirements for the particular part or product, and require the cooperation of Mechanical Engineering, Manufacturing Engineering, and Design Drafting. Tolerances may be specified for each individual dimension, or covered by one of the general tolerance notations.

General tolerances, which are made applicable unless otherwise specified, are selected to apply to the majority of dimensions on a drawing. The use of general tolerances saves drawing time by eliminating the need to indicate the tolerance of each dimension. General tolerances are never used where precision is required, but only when there is a need to establish a limit. Digital provides a 1-, 2-, and 3-place general tolerance format, and two dimension-sensitive general tolerance formats.

Dimensions shown without tolerances are controlled by the general tolerance notes in the title block of the drawing. The preprinted title blocks provide a selection of the general tolerances most commonly used by Digital, and require that the one being used be indicated. Figure 5 shows the standard inch title block; the standard metric title block is shown in Figure 6. The formats of both of these title blocks are controlled by *DEC STD 013-0 Standard Engineering Drawing - Formats and Forms*.

Formats created by computer-aided design (CAD) techniques are permitted to display only the general tolerances selected for use, provided the format is authorized by DEC STD 013-0.

Use of a standard Digital title block (Figure 5 or 6) requires that the designer select the general tolerance range that most appropriately fits the functional requirements of the part or assembly and the machine, method, or process used to fabricate or assemble the part without impairing its function or unnecessarily increasing its cost. The standard title blocks provide the following four tolerance options:

a. The first option is the use of the 1-, 2-, and 3-place decimal block (not available with the metric title block). When this option is selected, the other tolerance options should be crossed out, as shown in Figure 7.

b. The second and third options are choosing either of the dimension-sensitive tolerance ranges.

c. The fourth option is to use the blanks to specify any desired range. The specified range can be made dimension-sensitive by choosing tolerances to provide approximately constant accuracy, where the accuracy is the ratio of the tolerance to the dimension. If option 2, 3, or 4 is chosen, option 1 must be crossed out to make the drawing acceptable for release, as shown in Figure 8.

Figure 5: Standard Digital U.S. Customary (Inch) Title Block

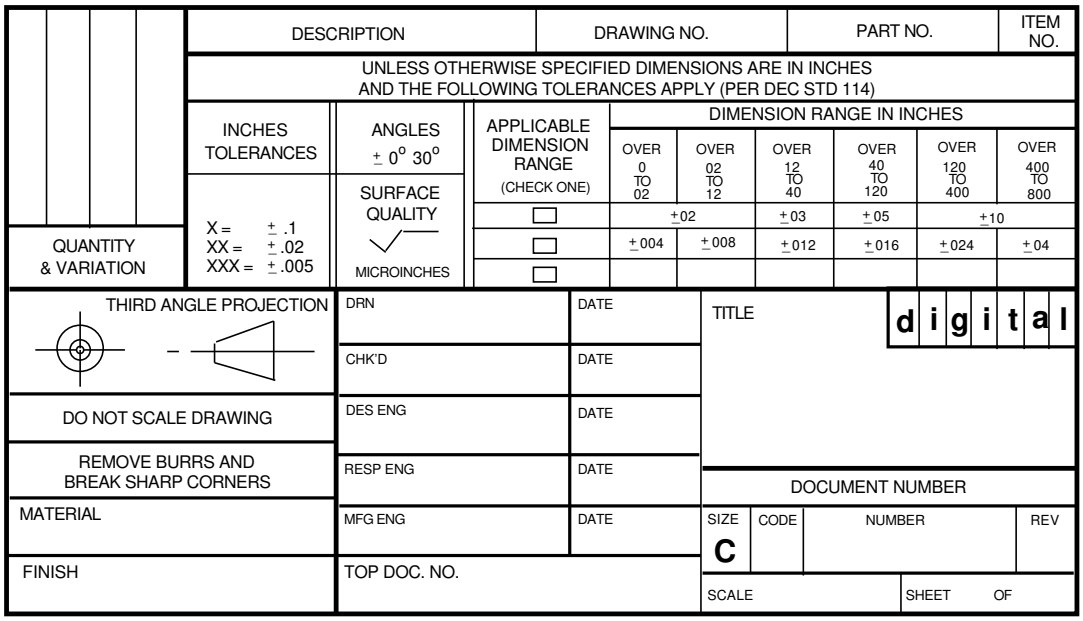


Figure 6: Standard Digital Metric Title Block

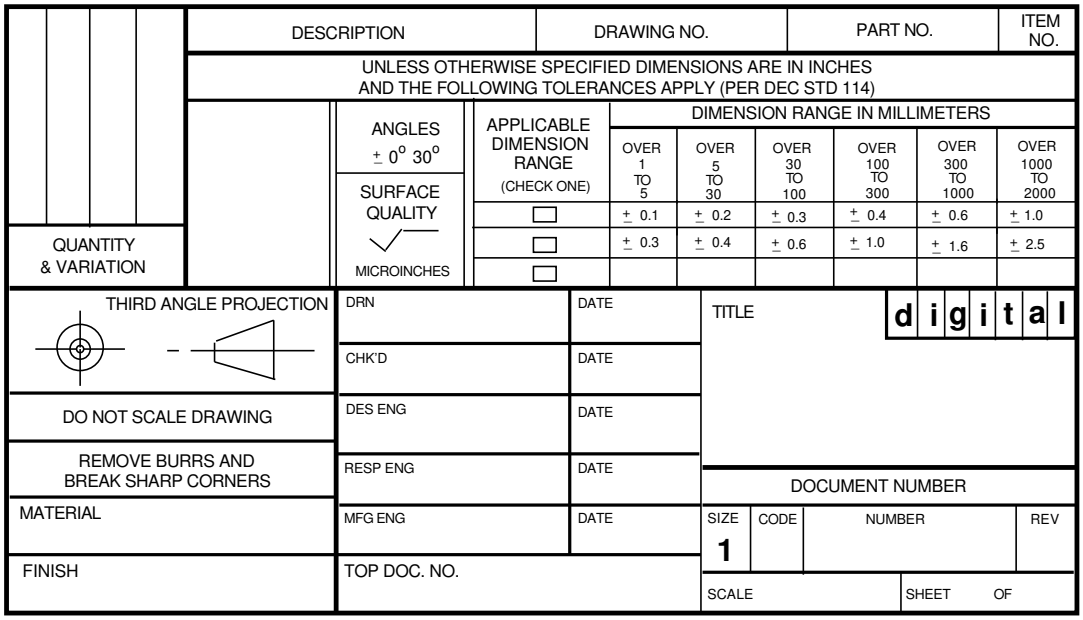


Figure 7: First Option Selected

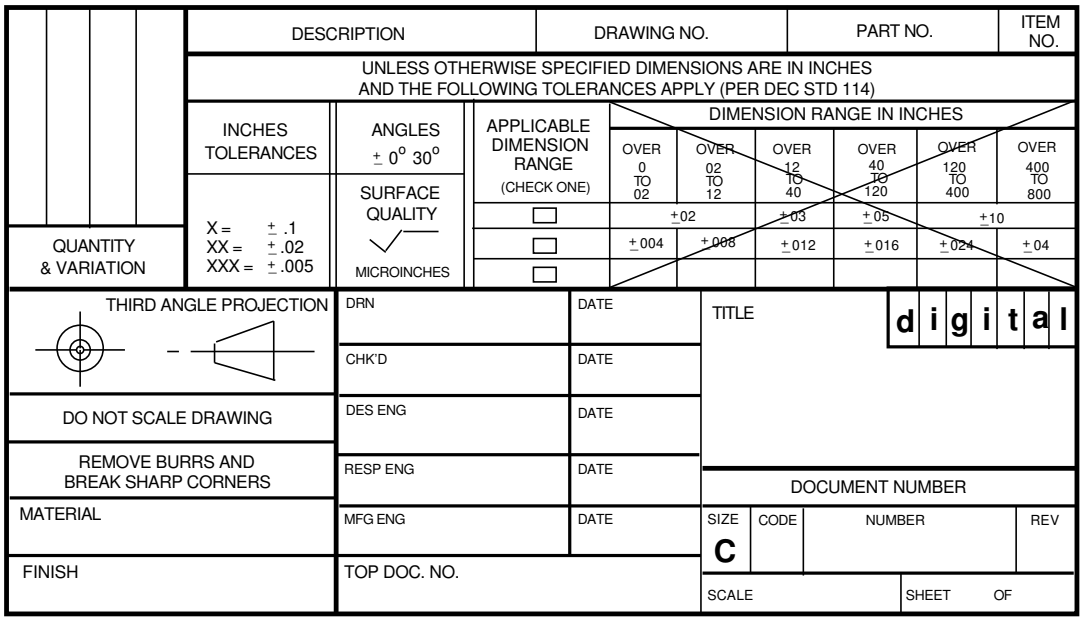
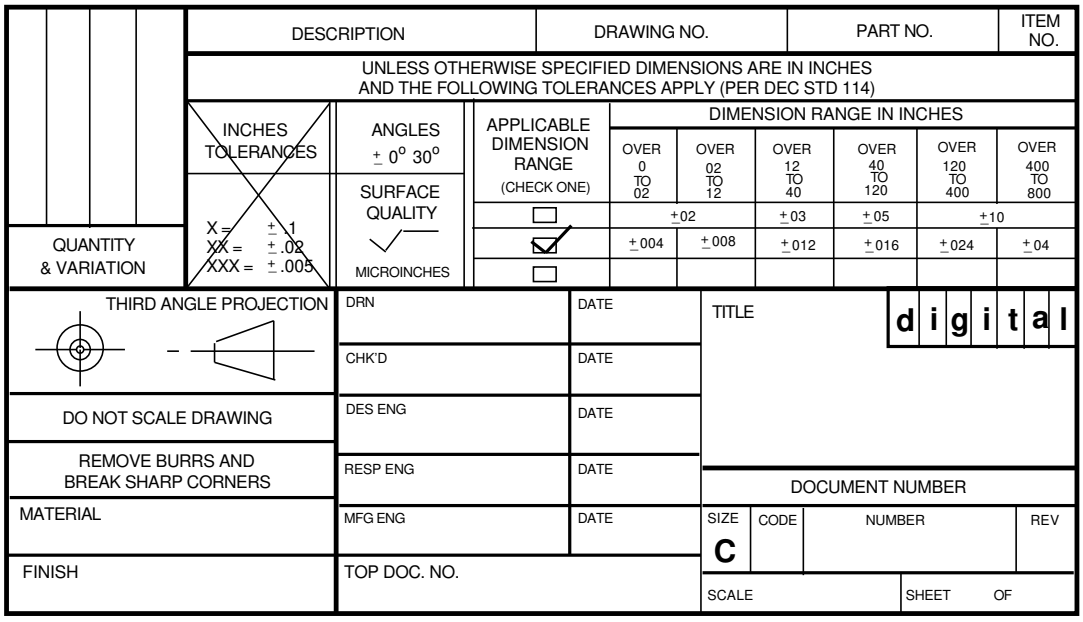


Figure 8: Third Option Selected



8 CONVERTING INCH DIMENSIONS TO METRIC

Conversion of inch dimensions into metric is done in one of two ways, depending on the requirements. Both methods start with precise conversion; the nearest value method then rounds to the nearest value at the same accuracy, while the tighter tolerance method rounds inside the original tolerance.

The following rules should be used to round off the nearest value:

a. Do not change the last digit retained (last digit) if the digit beyond it (next digit) is less than 5.

b. If the next digit is greater than 5, increase the last digit by 1.

c. If the next digit is 5, make the last digit even, adding 1 if it is odd.

To round to a tighter tolerance, truncate the upper limit and round up the lower limit. The following examples show both methods.

15.79 – 15.36 inches	= 401.066	– 390.144	millimeters	(exact)
	= 401.07	– 390.14	millimeters	(nearest)
	= 401.06	– 390.15	millimeters	(tighter)

9 CONVERTING FRACTIONS TO DECIMAL

When a drawing revision is required as part of an ECO to an existing part, all dimensions and tolerances should be checked for conformance to the requirements of this standard. Any conflicts should be considered by the responsible engineer, for inclusion in the ECO.

The following rules should be used to round off decimals that are too long:

a. Do not change the last digit retained (last digit) if the digit beyond it (next digit) is less than 5.

b. If the next digit is greater than 5, increase the last digit by 1.

c. If the next digit is 5, make the last digit even, adding 1 if it is odd. Table 2 shows the result of 2-, 3-, and 4-places for common fractions from 1/64 to 63/64.

Decimal Equivalents of Common Fractions

4ths	8ths	16ths	32nds	64ths	To 4 Places	To 3 Places	To 2 Places
				1/64	.0156	.016	.02
			1/32		.0312	.031	.03
				3/64	.0469	.047	.05
		1/16			.0625	.062	.06
				5/64	.0781	.078	.08
			3/32		.0938	.094	.09
				7/64	.1094	.109	.11
	1/8				.1250	.125	.12
				9/64	.1406	.141	.14
			5/32		.1562	.156	.16
				11/64	.1719	.172	.17
		3/16			.1875	.188	.19
				13/64	.2031	.203	.20
			7/32		.2188	.219	.22
				15/64	.2344	.234	.23
1/4					.2500	.250	.25
				17/64	.2656	.266	.27
			9/32		.2812	.281	.28
				19/64	.2969	.297	.30
		5/16			.3125	.312	.31
				21/64	.3281	.328	.33
			11/32		.3438	.344	.34
				23/64	.3594	.359	.36
	3/8				.3750	.375	.38
				25/64	.3906	.391	.39
			13/32		.4062	.406	.41
				27/64	.4219	.422	.42
		7/16			.4375	.438	.44
				29/64	.4531	.453	.45
			15/32		.4688	.469	.47
				31/64	.4844	.484	.48
1/2					.5000	.500	.50
				33/64	.5156	.516	.52
			17/32		.5312	.531	.53
				35/64	.5469	.547	.55
		9/16			.5625	.562	.56
				37/64	.5781	.578	.58
			19/32		.5938	.594	.59
				39/64	.6064	.609	.61
	5/8				.6250	.625	.62
				41/64	.6406	.641	.64
			21/32		.6562	.656	.66
				43/64	.6719	.672	.67
		11/16			.6875	.688	.69
				45/64	.7031	.703	.70
			23/32		.7188	.719	.72
				47/64	.7344	.734	.73
3/4					.7500	.750	.75
				49/64	.7656	.766	.77
			25/32		.7812	.781	.78
				51/64	.7969	.797	.80
		13/16			.8125	.812	.81
				53/64	.8281	.828	.83
			27/32		.8438	.844	.84
				55/64	.8594	.859	.86
7/8					.8750	.875	.88
				57/64	.8906	.891	.89
			29/32		.9062	.906	.91
				59/64	.9219	.922	.92
		15/16			.9675	.938	.94
				61/64	.9531	.958	.95
			31/32		.9688	.969	.97
				63/64	.9844	.984	.98
4/4					1.0000	1.000	1.00

APPENDIX A REFERENCED DOCUMENTS

A.1 EL-Class Digital Documents

EL-Class Number	Document Title
EL-00013-00	*DEC STD 013-0 Standard Engineering Drawing Formats and Forms - Introduction*
EL-00015-00	*DEC STD 015-0 Abbreviations and Units of Measurement*
EL-00182-00	*DEC STD 182-0 Engineering Documentation Acceptance Criteria*

Use VTX SMC to order copies of EL-class documents from Standards and Methods Control. Send distribution questions to JOKUR::SMC or call DTN: 234-4423.

A.2 Other Documents

Document Number	Document Title
-	*Geometric Dimensioning and Tolerancing - A Working Guide by Lowell Foster*
B46.1-1978	*Surface Texture, Including Surface Roughness, Waviness and Lay*
B87.1-1965	*Decimal Inch*
B94.6-1981	*Knurling*
Y14.2M-1979	*Line Conventions and Lettering*
Y14.5M-1982	*Dimensioning and Tolerancing*
Y14.6AM-1981	*Screw Thread Representation - Metric Supplement*
Y14.7.1-1971	*Gear Drawing Standards - Part 1 For Spur, Helical, double Helical and Rack*
Y14.7.2-1978	*Gear and Spline Drawing Standards Part 2 - Beveland Hypoid Gears*
Y4.6-1978	*Screw Thread Representation*

A.3 Ordering Information

Following are the sources for documents not available from Standards and Methods Control.

- ANSI Documents
  American National Standards Institute (ANSI),
  1430 Broadway, New York, NY 10018,
  Telephone (212) 354-3300 x479
- Geometric Dimensioning and Tolerancing
  Addison-Wesley, Jacob-Way,
  Reding, MA 01867

```
+-------------------------------------------------------------------+
|                        READER COMMENTS                            |
|                                                                   |
| Your  comments  and suggestions  will help  Standards and Methods |
| Control improve their services and documents.                     |
+-------------------------------------------------------------------+
```

Did you request this document? _____ If so, did it arrive within a satisfactory period of time? _____ Please comment.

What are your impressions of this document? Consider format, organization, completeness, readability, and illustrations.

————————FOLD ON THIS LINE———————————————————— -

Did you find technical or clerical errors in this document? If so, please specify the page number(s) and the error(s).

Are the instructions for the update package clear? _____
Was an index available? _____ If not, is one needed? _____
Do you have other suggestions for improving this document?

The following information is optional:

Name _____ Mailstop _____

Department _____ Node _____

Send your comments to JOKUR::PROJECTS, or fold, staple, and send this page through interoffice mail to:

```
+------------------------------+
|       READERS' COMMENTS      |
| STANDARDS AND METHODS CONTROL |
|          NRO4/D4             |
+------------------------------+
```